Applied Mechanics and Materials Vol 70 (2011) pp 201-206 *Online: 2011-08-18*
© (2011) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/AMM.70.201

Numerical simulation and Testing of water impact of structural attachment elements of a reusable thermal protection system

Dhanasekaran.M.P.[1,a], Balamurali.A.G.[1,b], Sundararajan.T.[1,c],
Jothi Ramalingam.A.[1,d], Dileep.R. [1,e], Sankaranarayanan.B.[1,f], Mohan.M.[1,g]

[1]Vikram Sarabhai Space Centre (VSSC)

Indian Space Research Organization (ISRO), Trivandrum, India – 695 022.

[a]mp_dhanasekaran@vssc.gov.in, [b]ag_balamurali@vssc.gov.in, [c]t_sundararajan@vssc.gov.in,
[d]jothi_ramalingam@vssc.gov.in, [e]r_dileep@vssc.gov.in, [f]b_sankaranarayanan@vssc.gov.in,
[g]m_mohan@vssc.gov.in

Keywords: Impact testing, Reusable Thermal Protection System (TPS), hot structure, structural attachments, strain, SRE

Abstract: Multiple mission reuse capability has become extremely important, towards reducing costs of space transportation. Carbon / Carbon (C/C) composites are well proven, functionally, for repeated use in re-entry missions. A re-entry capsule with sphere-cone-flare external shape, currently under realisation, will fly with C/C Thermal Protection System (TPS) in its peak heating region. The biggest challenge in design of such a reusable hot structure TPS is the management of thermo-structural loads. Differential Coefficient of Thermal Expansion (CTE) is the main cause of stress on the structural assembly elements. A set of flexible super alloy attachment brackets have been configured to take care of this differential thermal expansion of various TPS elements. The brackets also have to survive the impact transient load, on splashdown. This load was estimated using explicit non linear Finite Element method by considering the whole structure a rigid body. A separate FE model with actual stiffness of the structural attachments and the hot structure was generated, to predict the stresses caused by the load. In order to demonstrate the margins and survivability of the assembly, as a whole, a water impact test with actual qualification model of the assembly was carried out in a 10 m deep shock tank. The test also helped to validate the prediction. Considering factors such as cost, time and process constraints involved in realising C/C TPS for the test, it was decided to replace the same with an equivalent structure that satisfied all design and functional requirements. The test article was dropped vertically to simulate an impact velocity of 12 m/s and was adequately instrumented with accelerometers and strain gauges. The test results correlate reasonably well with the prediction.

Introduction

The primary function of a Thermal Protection System (TPS) in a reentry mission is to protect the underlying structure, electronic packages and payloads from intense aerodynamic heating. Several types of thermal protection systems have been used worldwide of which ablative type and radiative type are common. In order to reduce the costs of space transportation, reusability of systems have become increasingly vital. Carbon / Carbon (C/C) hot structure TPS has been successfully flown in peak heating zones of reusable space re-entry vehicles, such as the space shuttle, many times over.

Space capsule Recovery Experiment (SRE) is a sphere-cone-flare shape reentry capsule that was successfully flown [1] and recovered with ablative TPS in the fore end region and ceramic tiles in the cone and flare regions (Fig.1). The second flight of this capsule will have C/C TPS in the nose region in place of ablative TPS. The capsule has a base diameter of 2.0 m and a height of 1.67 m. It will be placed in a 635 Km Sun Synchronous Polar Orbit (SSPO). It will carry out microgravity experiments before it is deorbited for reentry. On reentry, the velocity of the capsule will initially be reduced by aerodynamic braking and later by a series of parachutes. It will impact on the Bay of Bengal with a velocity 12 m/s.

All rights reserved. No part of contents of this paper may be reproduced or transmitted in any form or by any means without the written permission of Trans Tech Publications, www.ttp.net. (ID: 152.14.136.77, NCSU North Carolina State University, Raleigh, USA-12/05/15,19:48:12)

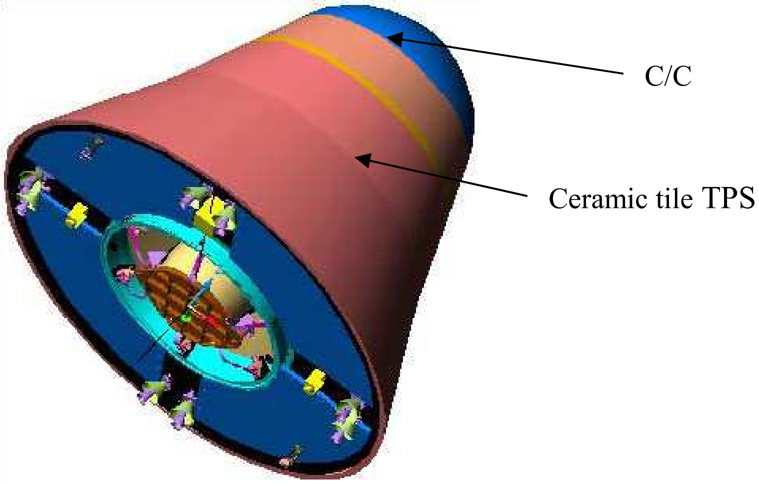


Fig. 1 SRE Capsule configuration

Unlike the back wall temperature of ablative systems, the back wall temperature of C/C TPS would typically be very high. Since the C/C TPS cannot be directly attached to the underlying structure, a number of flexible super alloy brackets (Fig.2) connect it with the underlying structure. These brackets that experience various types of loading during different phases of the mission would accommodate differential expansion of various elements of the assembly, thereby reducing the stress induced in C/C TPS. The predominant loads are thermo-structural loads during the reentry phase and a large transient load on impact. This paper focuses on the method used for estimating the impact loads and validation of the same through experimental testing.

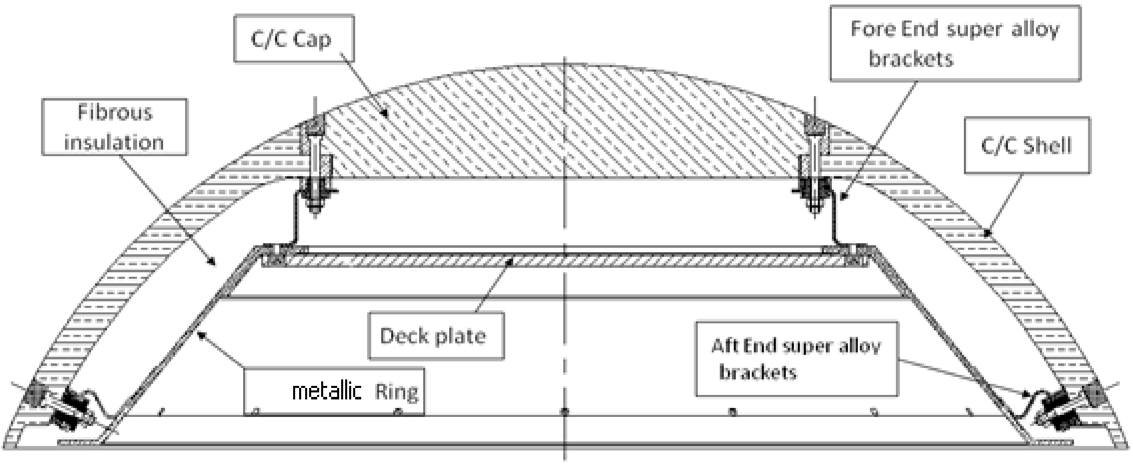


Fig.2 Configuration of Carbon / Carbon TPS

Numerical Simulation

The water impact of the capsule is a fluid-structure interaction problem. A detailed explanation of the approach used to carry out water impact analysis using explicit transient dynamic analysis code is given in Ref. [2]. A typical finite element idealisation of this problem is shown in Fig.3. The explicit dynamic analysis code provides two solving techniques viz. the Lagrangian and the Eulerian. The code can either use only one, or both, or can couple the two types to define an interaction. The Lagrangian method is the most common finite element solution technique for engineering applications. The Eulerian solver is most frequently used for analyses of fluids or materials that undergo very large deformations.

In the Lagrangian solver, grid points are fixed to locations on the body being analyzed. As the body deforms, the grid points move with the material and the elements distort. Also, it calculates the motion of elements of constant mass.

In the Eulerian solver, the grid points are fixed in space and the elements are simply partitions of the space defined by connected grid points. The material of a body under analysis moves through the Eulerian mesh; the mass, momentum, and energy of the material are transported from element to element. The Eulerian solver calculates the motion of a material through elements of constant volume. The fluid-structure interaction problem can be modeled using coupling algorithm that invokes the interaction between Eulerian and Lagrangian meshes.

Finite Element (FE) Model

For the present study a rigid model of the test article was used. The outer surface of the capsule was developed using Lagrangian shell elements. The total mass and mass moment of inertia of the module was lumped at its Centre of Gravity. The water and the air surrounding the capsule are modeled using 8 noded Eulerian, multi-material solid elements. Water and air properties are defined as multi-material hydrodynamic materials with no strength.

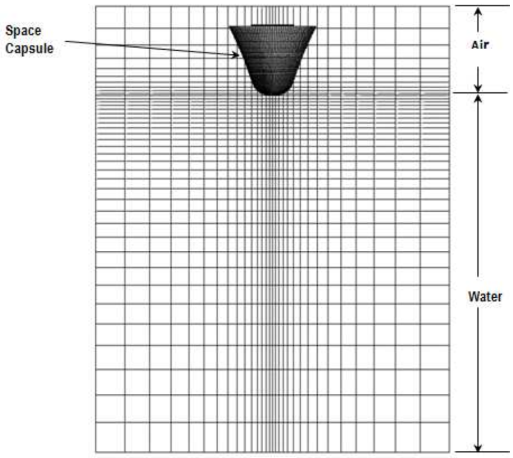


Fig.3 Idealization of the Water Impact Problem

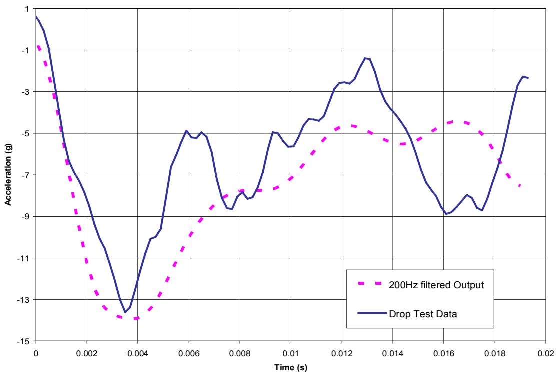
Air is modeled as void and polynomial equation of state is used to define water. Sensitivity of water mesh is one of the key factors that determine accuracy of the results obtained from water impact analysis. So the challenge, initially, in carrying out a water impact analysis was the development of a converged water mesh.

To build a converged water mesh, water impact analyses of spherical bodies were carried out. Results from these analyses were compared with closed form solutions available for rigid spheres. The analytical values of rigid sphere subjected to water impact were estimated based on the closed form solution given in Ref. [3]. The test article considered has a spherical nose cap of 0.5m radius and impacts the water in nose down condition. A good estimate of the maximum impact deceleration for this configuration can be calculated using the closed form solution for a 0.5m radius sphere. Impact analyses of rigid spheres were carried out iteratively using different mesh sizes to arrive at a converged mesh size. This mesh would then give a deceleration value that would matches closely with that obtained by solving the closed form solution.

Water Impact Analysis Results

Water impact analysis of the test article was carried out for an impact (nose-down) velocity of 12m/s, in the vertical direction. The horizontal component was assumed to be zero. The analysis was carried out to estimate the peak deceleration and depth of penetration. Fig.4 shows the time history of rigid body deceleration due to water impact. The raw output has numerical oscillations, an inherent drawback of explicit transient dynamic analysis involving fluid-structure interaction. A 200Hz low pass SAE filter was used to remove these oscillations. The peak deceleration obtained was 14.94g.

Since only the spherical cap of the capsule would be in contact with water at the instant of peak deceleration, the peak value can be compared with the closed form solution for a sphere of 0.5m radius and impact mass. The closed form value of sphere is computed based on the formulations developed in Ref. [3]. The peak value obtained for sphere is 14.5g against an FE value of 14.94g .



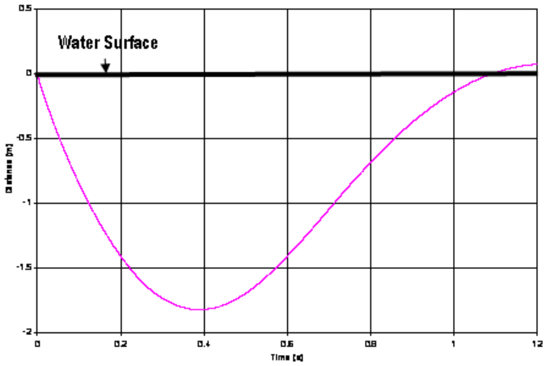
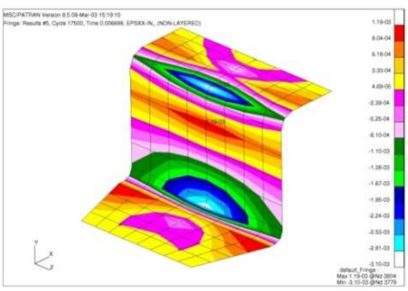


Fig.4 Deceleration of the Capsule Fig.5 Displacement of the Capsule Nose Tip

Fig.5 shows the displacement time history of the nose tip from the water surface. It can also be observed that the analysis captures the sinking of the capsule and subsequently, the rebound.

Strains

One of the objectives of the test was to estimate the strain at critical locations and compare it with theoretical predictions. The critical locations identified were the fore end and aft end brackets apart from the metallic ring. A separate finite element model of the test article was developed for computation of strains. The forward structure of the capsule was modeled as a flexible structure with appropriate properties. The acceleration profile obtained from the impact analysis was applied as equivalent pressure load on the nose of the article. The area of application of the pressure was computed based on the depth of penetration at the instant of peak deceleration. Four out of sixteen Fore and Aft end brackets (Each 90º apart, from one another) were identified and instrumented with strain gauges at the top and at the bottom, both on the inside and on the outside. Fig. 6 and Fig.7 show the typical stress contour for the fore and aft end brackets respectively at the instant of peak deceleration.



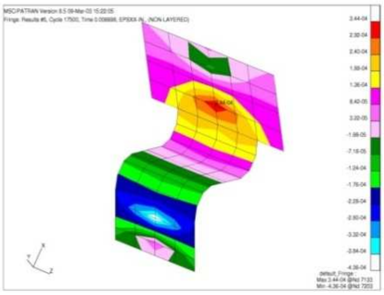


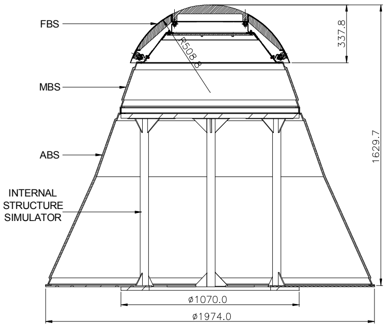
Fig.6 Stress contour for Fore End brackets Fig.7 Stress contour for Aft End brackets

Need of test & Objective

From the dynamic analysis of C/C TPS assembly, it is seen that the fore end brackets experience a large stress on impact, for a few milliseconds. Therefore, it was mandatory to demonstrate the adequacy of design for this impact load, through appropriate testing. Since it was not feasible to simulate this impact transient load through conventional structural tests, a water impact test was carried out. The objectives of the test were to qualify the structural attachment elements, demonstrate the survivability of these elements including its interfaces for impact transient loads and validate the numerical simulation.

Test setup & Instrumentation

The test hardware consisted of 4 subassemblies (Fig. 8): Forward Base Structure (FBS), Middle Base Structure (MBS), Aft Base Structure (ABS) and Internal Structure Simulator, similar to that of the flight structure. The FBS comprised flight equivalent components of structural attachment elements along with metallic structure simulators of C/C TPS. The test article was dropped vertically from a crane, from a height of 7.4 m (Fig. 9) to simulate an impact velocity of 12 m/s. The mass of test article was 536 kg to simulate the impact mass. Separation of the capsule from the crane would make use of a rope cutting system used in similar tests, earlier. Test was carried out in a Shock tank facility of dimensions 15 m x 10 m x 10 m (depth). 32 strain gauges were bonded to the super alloy brackets and 4 to the Aluminium structure to help deduce the stresses. All strain channels were quarter bridge channels with three-wire configuration.



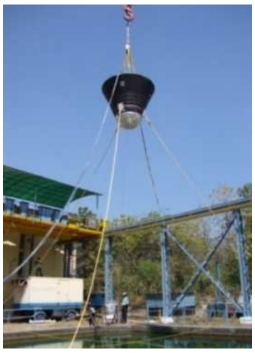


Fig. 8 Test Article Details

Fig. 9 Test set up

Two National Instruments SCXI based 24 channels Data AcQuisition Systems (DAQ) were used: one as prime and the other as redundant. For measuring the impact deceleration, velocity and depth of penetration of module, two sets of 3 axis accelerometers were provided; One connected to the prime and the other to the redundant. Provision for measuring 18 strains was given in the prime system and for rest in the redundant one. This scheme was worked out to ensure that even due to failure of one acquisition system, meaningful test data could be acquired. A separation connector triggered both DAQ systems simultaneously as the module fell from the required height. The connector mating status was acquired in both the DAQ systems for time reference. The sampling rate was 5 kHz and filter setting was 1 kHz. Duration of data acquisition was 7 seconds.

Test results & Discussion

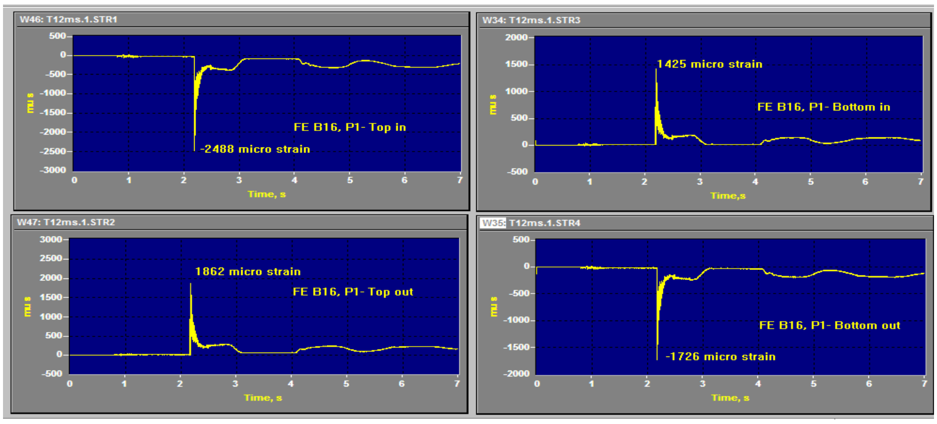


Fig.10 Measured strain in FE bracket

The measured peak rigid body deceleration was 16.6g against the prediction of 14.5g. The impact velocities derived from the acceleration data were 11.99 m/s and 11.88 m/s corresponding to the prime and redundant accelerometers respectively. A typical strain history of a single bracket is shown in Fig. 10. The peak strains from the test are compared in Table 1 with that obtained from prediction. The measured strains match very well with the predicted values for Fore End (FE) brackets. However, the match for the Aft End (AE) brackets is not as good as that for the Fore End brackets. A possible reason for the mismatch could be the prominent *strain gradient* observed from the analysis in the vicinity of the strain gauge locations. In the presence of this gradient, any small misplacement of the strain gauges from the desired locations could result in the same.

Table 1 Comparison of Strain

Location	SG No.	Maximum Strain [$\mu\varepsilon$]	
		Test	Prediction
FE Bracket 16 (P1-) Top In	1	-2488	-1969
FE Bracket 16 (P1-) Top Out	2	+1862	+1636
FE Bracket 16 (P1-) Bottom In	3	+1425	+1032
FE Bracket 16 (P1-) Bottom Out	4	-1726	-1687
AE Bracket 16 (P1-) Top In	17	-700	-158
AE Bracket 16 (P1-) Top Out	18	+482	+171
AE Bracket 16 (P1-) Bottom In	19	+312	+40
AE Bracket 16 (P1-) Bottom Out	20	-478	-51

Conclusion

Explicit transient dynamic analysis finite element code was used to predict water impact characteristics. The peak deceleration computed using FE analysis shows a very good comparison with that deduced from closed form solutions and the water impact test. Importantly, the predicted strains on the fore end brackets - critical to the design- matches reasonably well with the measured values.

Acknowledgements

Authors wish to acknowledge the contributions of Mr. AVS Chari, Mr. Kumar Subham, Mr. Ashok Kumar Singh, Mr. A. K. Sinha, Mr. Vijaykumar, Mr. Abhinav Rukela, Mr. Arun Zachariah, Mr. Sudhir S. Nair, Mr. C.S.Krishnakumar, Mr. Sanjay Kumar Singh, Mr. Dhandhukiya I J, Mr. Saji V.J., Mr. Santhosh J.N., Mr. Ritesh Pillai and Mr. S.S. Murali.

References

[1] A. Subramoniam, Madanlal, N.C. Bhat, N.P. Giri, Koshy Mammen, A. Krishnan and P. Sunil : *Space Capsule Recovery Experiment Project – Mission Overview*, IAC–07: D2.2.04, 58th International Astronautical Congress (2007).

[2] Wang, John T and Lyle, Karen H. : *Simulating Space Capsule Water Landing with Explicit Finite Element Method*, AIAA 2007-1779, 48th AIAA/ASME/ASCE/AHS/ASE Structures, Structural Dynamics, Material Conference, Honolulu, Hawaii (2007)

[3] Hirano, Yoichi and Miura, Koryo : *Water Impact Acceleration of Axially Symmetric Bodies*, Journal of Spacecraft and Rockets, Vol. 7, No. 6 (1970)

Advances in Experimental Mechanics VIII
10.4028/www.scientific.net/AMM.70

Numerical Simulation and Testing of Water Impact of Structural Attachment Elements of a Reusable Thermal Protection System
10.4028/www.scientific.net/AMM.70.201

DOI References

[3] Hirano, Yoichi and Miura, Koryo : Water Impact Acceleration of Axially Symmetric Bodies, Journal of Spacecraft and Rockets, Vol. 7, No. 6 (1970).
doi:10.2514/3.30037